Chinese Journal of Aeronautics, 2013,26(2): 287–293

Chinese Society of Aeronautics and Astronautics
& Beihang University

Chinese Journal of Aeronautics

cja@buaa.edu.cn
www.sciencedirect.com

Investigation of thermal protection system by forward-facing cavity and opposing jet combinatorial configuration

Lu Haibo [a,b,*], Liu Weiqiang [a]

[a] *Science and Technology on Scramjet Laboratory, National University of Defense Technology, Changsha 410073, China*
[b] *Nanjing Artillery Academy, Nanjing 211100, China*

Received 15 November 2011; revised 22 February 2012; accepted 13 March 2012
Available online 6 March 2013

KEYWORDS

Aerodynamic heating;
Forward-facing cavity;
Hypersonic flow;
Opposing jet;
Thermal protection system

Abstract This paper focuses on the usage of the forward-facing cavity and opposing jet combinatorial configuration as the thermal protection system (TPS) for hypersonic vehicles. A hemisphere-cone nose-tip with the combinatorial configuration is investigated numerically in hypersonic free stream. Some numerical results are validated by experiments. The flow field parameters, aerodynamic force and surface heat flux distribution are obtained. The influence of the opposing jet stagnation pressure on cooling efficiency of the combinatorial TPS is discussed. The detailed numerical results show that the aerodynamic heating is reduced remarkably by the combinatorial system. The recirculation region plays a pivotal role for the reduction of heat flux. The larger the stagnation pressure of opposing jet is, the more the heating reduction is. This kind of combinatorial system is suitable to be the TPS for the high-speed vehicles which need long-range and long time flight.

1. Introduction

Hypersonic vehicles such as interceptor missiles, re-entry vehicles, hypervelocity projectiles and hypersonic aircraft are designed to withstand severe heat loads. The scholars in thermal protection fields are always keeping their eyes on the design of high speed vehicles' thermal protection system. With the development of the spacecraft, new function requirements, such as reusable, are desired. Some traditional thermal protection techniques, for example, the ablation technique,[1] are difficult to satisfy these requirements. New techniques such as platelet transpiration,[2] heat-pipe,[3] thermal photovoltaic (TPV)[4] were used to the thermal protection system.

In 1921, a body containing a forward-facing cavity under a supersonic flow was introduced first by Hartmann.[5] It was used as a new technique for producing sound of high intensity and discrete frequency at that time, which was known as the "Hartmann whistle". Research efforts related to these ideas have been done by a number of researchers.[6] In 1959, Burbank and Stallings[7] reported this idea as a thermal protection technique for the nose-tip of hypersonic vehicles firstly. In recent years, attracted by its simple structure and excellent thermal

* Corresponding author at: Science and Technology on Scramjet Laboratory, National University of Defense Technology, Changsha 410073, China. Tel.: +86 731 84574177.
E-mail addresses: lhbboo@sohu.com (H. Lu), liuweiqiang_1103@163.com (W. Liu).

Peer review under responsibility of Editorial Committe of CJA.

ELSEVIER Production and hosting by Elsevier

1000-9361
http://dx.doi.org/10.1016/j.cja.2013.02.005

protection effect, many studies have done on it. Preliminary experiments, using an infrared (IR) camera, by Yuceil et al.[8] indicated that large diameter, shallow cavities (length-to-diameter ratio L/D between 0.15 and 0.35) created a stable "cool ring" in the vicinity of the sharp cavity lip, which means that the local temperature was lower than that of a simple spherical nose. Seiler et al.[9] researched on the thermal protection efficiency of forward-facing cavity at Mach number $Ma = 4.5$. An important result of their study is that the deepest cavity has the smallest heat flux. Saravanan et al.[10,11] investigated the effects of a forward-facing cavity on heat transfer and aerodynamic coefficients. Numerical simulation was carried out with steady-state flow assumption and had a good agreement with their tests in hypersonic shock tunnel HST2, at a hypersonic Mach number of 8.

In the early 1960s, opposing jet was reported as a thermal protection technique for the nose-tip of hypersonic vehicles and validating experiments were conducted.[12] More studies on opposing jet flow have been conducted in the 21st century. Aerodynamic heating reduction[13] due to opposing jet from the top of blunt body is experimentally and numerically investigated. Aerodynamic heating reduction due to opposing jet is proved to be quite effective at the nose of the blunt body by experiment. Detailed numerical investigation of the flow field indicates that the recirculation region plays an important role in reduction of aerodynamic heating. The effect of the ratio of stagnation pressure of opposing jet to free stream on the reduction of aerodynamic heating is investigated by Hayashi et al.[14,15] The experimental and numerical results showed that as the pressure ratio increased, the heat flux decreased at each point of the nose surface. The detailed influences of the free Mach number, jet Mach number and attack angle on the reduction of drag coefficient were studied by high precise simulation of Navier–Stokes equations.[16]

In the present study, the forward-facing cavity and opposing jet combinatorial TPS is investigated numerically. Some numerical results are validated by experiments. Remarkable aerodynamic heating reduction due to the combinatorial TPS in hypersonic flow field is revealed by detailed numerical simulation. Furthermore, the influence of the opposing jet stagnation pressure on thermal protection efficiency of the TPS is discussed.

2. Configuration of combinatorial TPS

The configuration of the forward-facing cavity and opposing jet thermal protection system is shown in Fig. 1. The nozzle exit of the opposing jet is located at the center of the base wall of the cavity and the diameter is 4 mm. The fluid medium for opposing jet is assumed as compression air. In order to compare with the results of experiment, the geometric configuration of the nose-tip is the same as that in Ref.[11] The depth of the cavity L is 24 mm and the diameter D is 12 mm, the diameter of the nose-tip bottom surface D_n is 51 mm.

3. Computation scheme

3.1. Governing equation

The 3-D Navier–Stokes equations is given by[17]

$$\frac{\partial U}{\partial t}+\frac{\partial E}{\partial x}+\frac{\partial F}{\partial y}+\frac{\partial G}{\partial z}=\frac{\partial E_v}{\partial x}+\frac{\partial F_v}{\partial y}+\frac{\partial G_v}{\partial z} \quad (1)$$

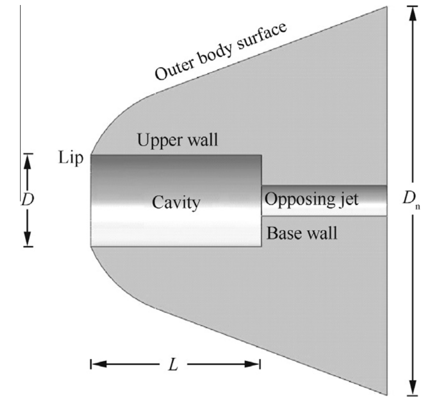


Fig. 1 Schematic of combinatorial TPS.

where (x, y, z) is the coordinate of the physical space, U the conservation variable, E, F and G are the inviscid terms, and E_v, F_v and G_v are the viscous terms. The k-ε turbulence model[18] is used in the simulation.

The convective terms are approximated using the advection upstream splitting method-DV (AUSM-DV) splitting method[19] and central difference method for the viscous terms. The lower–upper symmetric successive over relaxation (LU-SSOR) scheme[20] is used for the time integration.

3.2. Generation of grids

The three-dimensional boundary-fitted grids for the nose-tip with three kinds of TPS are generated by the Poisson equation.[21]

$$\begin{cases} \nabla^2_{(x,y,z)}\xi = P \\ \nabla^2_{(x,y,z)}\eta = Q \\ \nabla^2_{(x,y,z)}\varsigma = R \end{cases} \quad (2)$$

where (ξ, η, ζ) is the coordinate of the calculation space, and P, Q, R are the source items which control and regulate the refinement of the grid.[22,23]

The grid of simulation model (nose-tip with combinatorial TPS) on the symmetry plane and on the wall of the nose-tip is shown in Figs. 2 and 3.

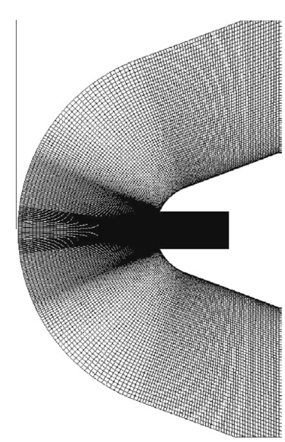


Fig. 2 Grid on symmetry plane.

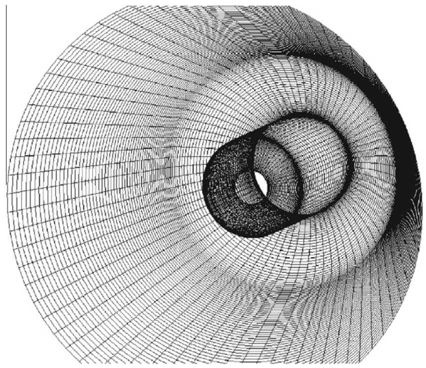


Fig. 3 Grid on nose-tip object surface.

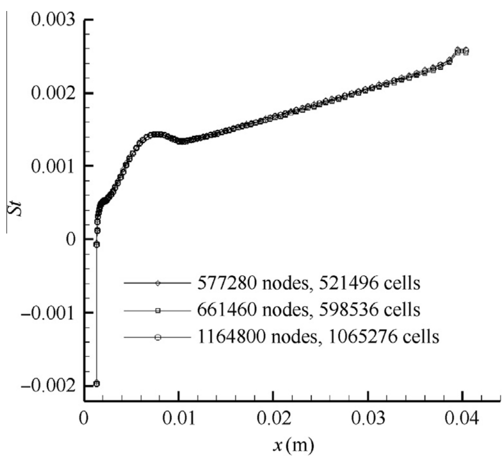


Fig. 4 Assessment of grid resolution study.

Fig. 4 shows the Stanton number St predicted by three different meshes. It is seen that there is no change in St, irrespective of grid size. These meshes produce very close matching throughout the model length. Finally, the total number of around 661460 nodes (grid points) have been chosen for the present study.

3.3. Boundary condition and numerical assumption

The flow conditions of the free stream and opposing jet are shown in Tables 1 and 2. The boundary condition of wall is assumed isothermal (wall temperature $T_w = 300$ K) and no-slip there. In order to obtain stable simulation results, a steady-state flow condition is assumed in simulation of nose-tip with forward-facing cavity (no opposing jet), which is different from the high speed cavity flow in reality.[6]

4. Numerical methodology validation

Stanton number based on the free stream condition is given by the expression:

$$St = \frac{q_w}{(T_{aw} - T_w)\rho_\infty c_{p\infty} u_\infty} \tag{3}$$

$$T_{aw} = T_\infty\{1 + \sqrt[3]{Pr}[(\gamma - 1)/2]Ma_\infty^2\} \tag{4}$$

Table 1 Flow condition of free stream.

Parameter	Quantity	Value
Ma_∞	Mach number	8
P_0 (Pa)	Stagnation pressure	1939211
$T_{0\text{-free}}$ (K)	Stagnation temperature	1955.14

Table 2 Flow condition of opposing jet.

Parameter	Quantity	Value
Ma_{opp}	Mach number	1
PR	Ratio of stagnation pressure of free stream to opposing jet	0.05, 0.10, 0.20
$T_{0\text{-opp}}$ (K)	Stagnation temperature	300

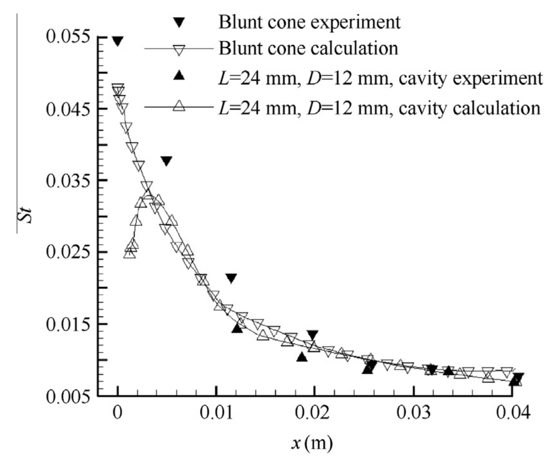


Fig. 5 Stanton number comparison.

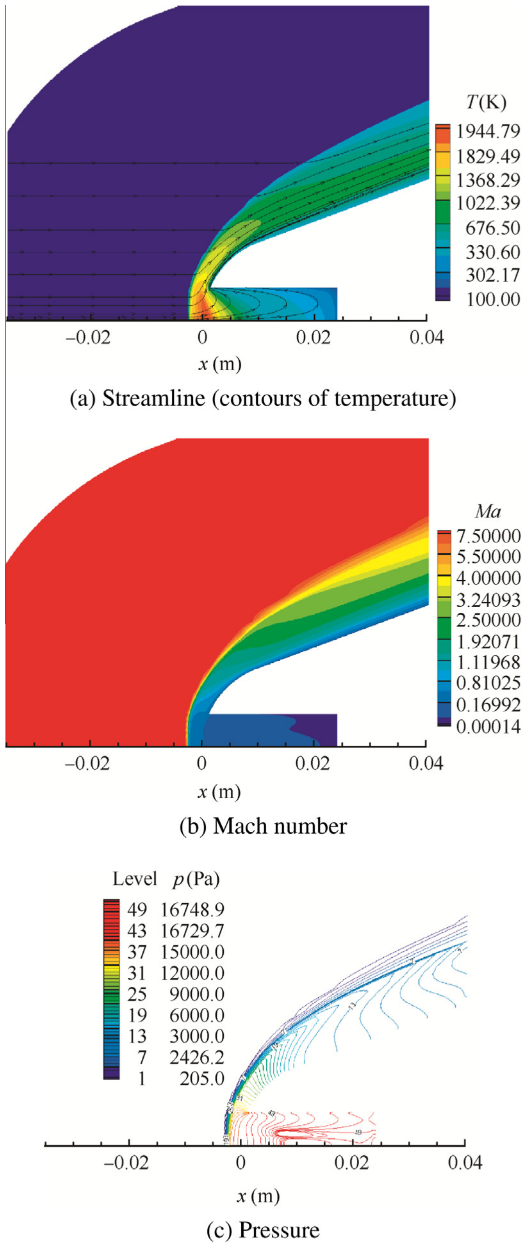
where q_w is the heat flux, T_{aw} the temperature of the thermal isolation wall, T_w the temperature of the outer body surface, ρ_∞ the density of the free stream, $c_{p\infty}$ the constant-pressure specific heat of the free stream, u_∞ the velocity of the free stream, T_∞ the static temperature of the free stream, Pr Prandtl number, and γ specific heat ratio.

The numerical and experimental[11] results of the Stanton number along the outer body surface of a traditional blunt cone and nose with forward-facing cavity (without opposing jet) is compared in Fig. 5. An agreement between numerical and experimental results is shown. The discrepancy comes from the assumption of simulation model, counting error and experimental measurement. Because of the cavity, it is hard to set the sensors and find data near the cavity lip. The numerical results show that an explicit "cool ring" phenomenon exists just outside of the sharp cavity lip. The forward-facing cavity configuration does well in cooling the nose especially near the stagnation area.

5. Results and discussion

5.1. Flow field

The streamline (with contours of temperature), Ma and pressure distribution of nose-tip with forward-facing cavity are

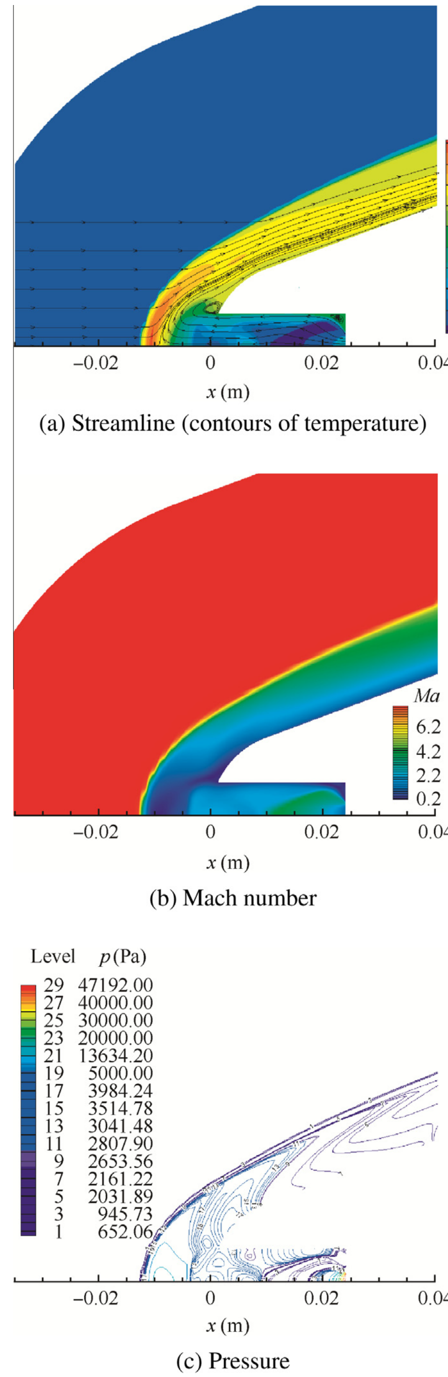


(a) Streamline (contours of temperature)

(b) Mach number

(c) Pressure

Fig. 6 Flow field of nose-tip with forward-facing cavity.



(a) Streamline (contours of temperature)

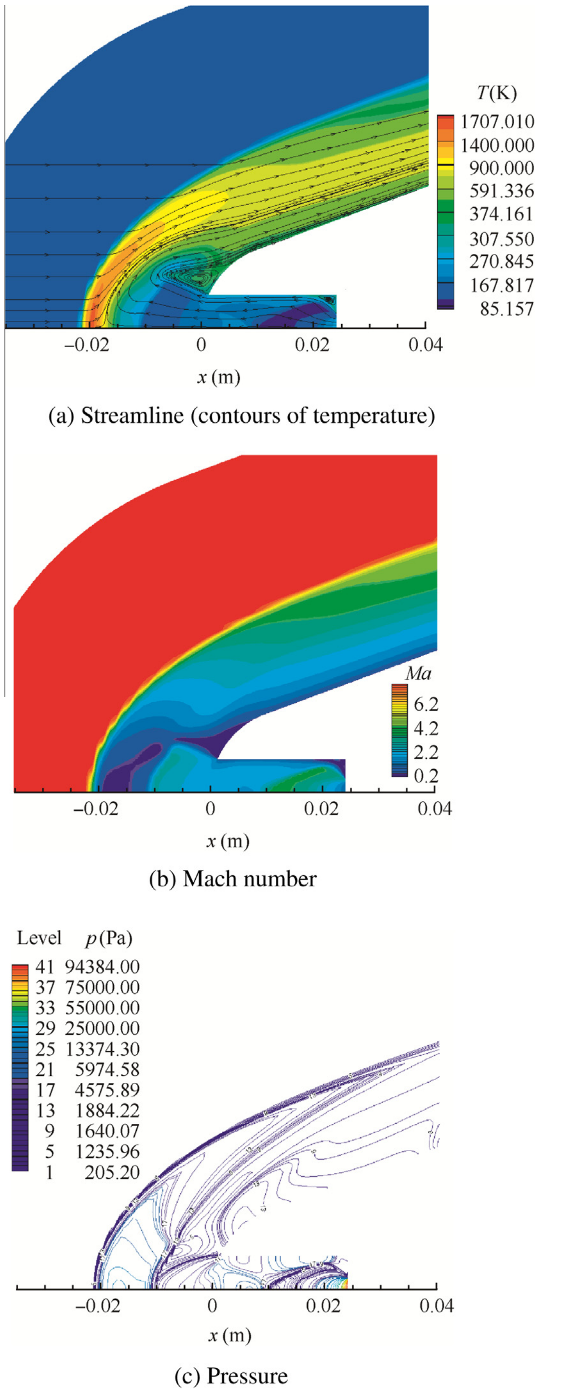
(b) Mach number

(c) Pressure

Fig. 7 Flow field of nose-tip with combinatorial TPS (PR = 0.05).

shown in Fig. 6, and of combinatorial TPS with PR 0.05, 0.10 and 0.20 are shown respectively in Figs. 7–9. From these figures, it is evident that after the opposing flow jets out from the nozzle, there is a rapid expansion of it, and a clear reflected wave is formed from the upper wall in the cavity. The more the opposing jet stagnation pressure is, the stronger the reflected wave is. Out of the cavity, a Mach disk is formed in order to balance the pressure of opposing jet and the pressure behind the detached shock wave. Opposing jet meets free stream and forms the interface. The free steam lets the jet layer reattach to body surface, then recirculation region is formed around the cavity lip and recompression shock wave is formed near the reattachment of the jet layer.

It can be seen from Fig. 7(a), Fig. 8(a) and Fig. 9(a) that there are two recirculation regions in the flow field. One is located at the corner of the cavity, which is formed by the opposing jet and the shape of the cavity bottom; the other is around the cavity lip. The higher the opposing jet stagnation pressure is, the stronger the resistance from opposing jet against free stream is, then, the larger the recirculation region is. As shown in the part (a) of all these figures, the highest temperature region is formed downstream the bow shock and in front of the cavity. With the opposing jet speed increasing, the location of it is further away from the cavity lip. In part (c), except the nose-tip with forward-facing cavity, there is no big change in

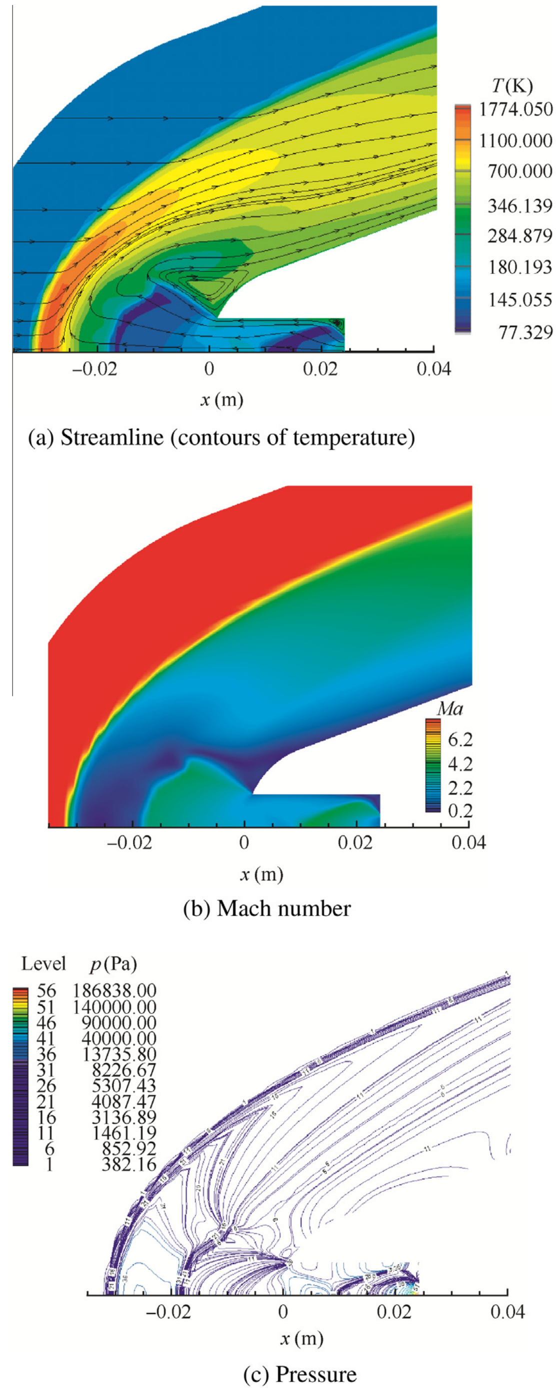


(a) Streamline (contours of temperature)

(b) Mach number

(c) Pressure

Fig. 8 Flow field of nose-tip with combinatorial TPS (PR = 0.10).



(a) Streamline (contours of temperature)

(b) Mach number

(c) Pressure

Fig. 9 Flow field of nose-tip with combinatorial TPS (PR = 0.20).

the pressure distributions and the main alteration is concentrated at the exit of the opposing jet.

The numerical simulation results of the combinatorial TPS flow field shows that the forward-facing cavity is just like a nozzle for the opposing jet. It forms a typical opposing jet flow in front of the nose-tip, so the fully developed flow field of this TPS should be a steady-state one. The harm to the aircraft control performance from the oscillating of the hypersonic cavity flow is avoided.

5.2. Aerodynamic force

The drag coefficient C_D is given by the expression:

$$C_D = F_D / \left(\frac{1}{2} \rho_\infty u_\infty^2 S_{\text{ref}} \right) \tag{5}$$

where F_D is the aerodynamic resistance, and S_{ref} the reference area which is the bottom surface of the nose-tip (diameter D_{n}).

Table 3 Drag coefficient of combinatorial conformation.

Parameter	Value			
PR	0	0.05	0.10	0.20
C_D	0.47698	0.37685	0.27390	0.23090

The drag coefficient C_D of the combinatorial conformation with different opposing jet speeds is shown in Table 3 (PR = 0 means no opposing jet). The drag coefficient decreases with the opposing jet stagnation pressure increasing. As shown in Figs. 7–9, the higher the opposing jet stagnation pressure is, the sharper the shape of the bow shock is, then, the smaller the aerodynamic resistance is.

5.3. Thermal protection efficiency

The *St* distributions along the outer body surface of nose-tip are shown in Fig. 10. Compared with the no opposing jet cases, the cooling efficiency of the combinatorial TPS is much better than the nose-tip with single forward-facing cavity. The opposing jet meets free stream, the free steam lets the jet layer reattach to body surface and the free stream is separated from the nose-tip by the jet layer. Because the free stream cannot attach the body surface directly, the aerodynamic heating is reduced. The low enthalpy opposing jet flow even causes a heat release at the lip of the nose (the *St* is negative at the lip, case PR = 0.10 and 0.20). Under the conditions of opposing jet flow $Ma = 1$, the aerodynamic heating reduction is increased with the opposing jet stagnation pressure increasing.

Along the outer body surface of the nose-tip, the flow changes from a low-speed expansion one around the hemisphere into a direct one along the cone. So, there is a change of heat flux tendency at the interface of the hemisphere and cone. With the opposing stagnation pressure increasing, this phenomenon becomes weak.

Along the back-end surface of the nose-tip, the heat flux of case PR = 0.05 is higher then the case PR = 0 (nose-tip with cavity only). As shown in Fig. 7(a), when PR = 0.05, the recirculation region forms out of the cavity and the scale of it is small by its relatively lower opposing pressure. This reflux

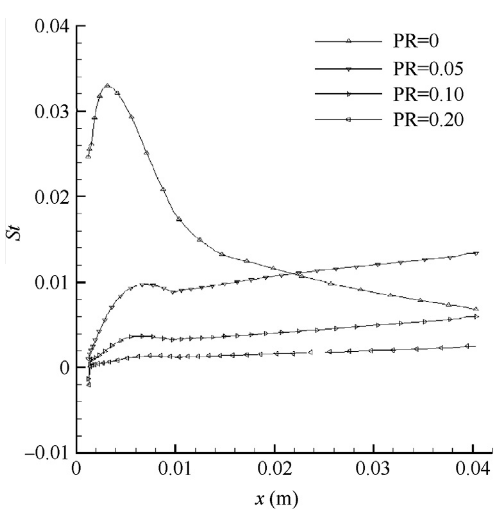


Fig. 10 *St* distributions.

causes a plentiful contact between the opposing flow and the high temperature gas in the maximum temperature region behind the bow shock. In this case, the mass flow rate of opposing flow is much smaller than the other two cases, and there is no enough low enthalpy gas to neutralize or isolate the influence of the high enthalpy gas. The low temperature opposing flow gas and the high temperature gas are mixed well. So, the temperature of the air flow along the back-end of the nose-tip is higher than other cases (see Figs. 6–9). The heat flux of case PR = 0.05 is higher then the case PR = 0, along the back-end of nose-tip. The recirculation region plays an important role for the reduction of heat flux.

It can be seen from the Fig. 10 that different thermal protection effects can be selected by choosing the opposing jet stagnation pressure. According to the flight condition, the opposing jet can be stopped when the aerodynamic heating is acceptable by single forward-facing cavity cooling. The fluid medium for opposing jet will be saved in this model. When the heating is intolerable with the flight speed increasing, the opposing jet is started with a suitable Mach number to get a new and stronger cooling efficiency.

6. Conclusions

(1) The forward-facing cavity and opposing jet combinatorial system is suitable to be the TPS for the hypersonic vehicles, which fly under various flow conditions and need long-range and long time flight.
(2) The forward-facing cavity can improve the aerodynamic heating environment of the nose-tip especially in the vicinity of the stagnation point. The combinatorial TPS has an excellent effect on cooling the nose-tip at each point. By adding of an opposing jet with a small stagnation pressure, the cooling efficiency of combinatorial configuration can be much better than a single cavity. But if the opposing jet pressure is too small (PR = 0.05 in the paper), the aerodynamic heating on the back-end of the nose-tip will be worse.
(3) The recirculation region plays an important role in the reduction of heat flux. The recirculation region has a close relationship with the opposing jet stagnation pressure. Different cooling efficiency can be selected by opposing jet stagnation pressure control. The larger the opposing jet stagnation pressure is, the better the cooling efficiency is. What's more, the situation of whether the opposing jet works or not can be controlled, too. When the opposing jet is stopped, the fluid medium for opposing jet will be saved and the aerodynamic heating will be reduced by the single cavity.
(4) The aerodynamic drag of the nose-tip is decreased by the combinatorial TPS. The drag coefficient is decreasing with the opposing jet stagnation pressure increasing. And this TPS can avoid the disadvantage of the aircraft control performance which is caused by the hypersonic cavity flow oscillating.

Acknowledgements

This work was co-supported by National Natural Science Foundation of China (No. 90916018); Research Fund for the

Doctoral Program of Higher Education of China (No. 200899980006).

References

1. Zheng TL, Zou JC, Yu B, Wang YH, Zhang H. Study on low density and heat-resistant ablative coating. *Chin J Aeronaut* 2005;**18**(4):372–7.
2. Zhang F, Liu WQ. Analysis of thermal wrinkling resulted from non-uniform temperature distribution in transpiration cooling formed platelets. *Acta Aeronaut Astronaut Sin* 2007;**28**(1):138–41 [Chinese].
3. Glass DE. Heat-pipe-cooled leading edges for hypersonic vehicles. NASA Langley Research Center; 2006.
4. Ye H, Geng X. The feasibility analysis of the application of TPV system in reentry. *Sci China Technol Sci* 2010;**41**(1):102–8.
5. Hartmann J. On a new method for the generation of sound-waves. *Phys Rev* 1922;**20**(6):719–27.
6. Engblom WA, Goldstein DB. Fluid dynamics of hypersonic forward-facing cavity flow. *J Spacecraft Rockets* 1997;**34**(4):437–44.
7. Burbank PB, Stallings RL. Heat-transfer and pressure measurements on a flat nose cylinder at a mach number range of 2.49 to 4.44. Report No.: NASA-TM-X-221; 1959.
8. Yuceil B, Dolling DS, Wilson D. A preliminary investigation of the Helmholtz resonator concept for heat flux reduction. Report No.: AIAA-1993-2742; 1993.
9. Seiler F, Srulijes J, Gimenez Pastor M, Mangold P. Heat fluxes inside a cavity placed at the nose of a projectile measured in a shock tunnel at mach 4.5. In: New results in numerical and experimental fluid mechanics VI; 2008. p. 309–16.
10. Saravanan S, Nagashetty K, Jagadeesh G, Reddy KPJ. Experimental investigation of heat transfer reduction using forward facing cavity for missile shaped bodies flying at hypersonic speed. In: 26th international symposium on, shock waves; 2007. p. 316–7.
11. Saravanan S, Jagadeesh G, Reddy KPJ. Investigation of missile-shaped body with forward-facing cavity at Mach 8. *J Spacecraft Rockets* 2009;**46**(3):577–91.
12. Warren CHE. An experimental investigation of the effect of ejecting a coolant gas at the nose of a bluff body. *J Fluid Mech* 1960;**8**(3):400–17.
13. Aso S, Hayashi K, Mizoguchi M. A study on aerodynamic heating reduction due to opposing jet in hypersonic flow. Report No.: AIAA-2002-0646; 2002.
14. Hayashi K, Aso S. Effect of pressure ratio on aerodynamic heating reduction due to opposing jet. Report No.: AIAA-2003-4041; 2003.
15. Hayashi K, Aso S, Tani Y. Numerical study of thermal protection system by opposing jet. Report No.: AIAA-2005-188; 2005.
16. Tian T, Yan C. Numerical simulation on opposing jet in hypersonic flow. *J Beijing Univ Aeronaut Astronaut* 2008;**34**(1):9–12 [Chinese].
17. Chen LZ. Numerical simulation and research of the unsteady separation flows around double-delta wings [dissertation]. Mianyang: China Aerodynamics Research and Development Center; 2008 [Chinese].
18. Tao WQ. *Numerical heat transfer*. 2nd ed. Xi'an: Xi'an Jiaotong University Press; 2001 [Chinese].
19. Wada Y, Liou MS. A flux splitting scheme with high-resolution and robustness for discontinuities. Report No.: AIAA-1994-0083; 1994.
20. Wang CY, Wang ZH, Yang XH. *Computational fluid dynamics and parallel algorithm*. Changsha: National University of Defense Technology Press; 2000 [Chinese].
21. Baker AJ. Numerical grid generation techniques. Report No.: NASA-CP-2166; 1980.
22. Thomas PD, Middelcoff JF. Direct control of the grid point distribution in meshes generated by elliptic equations. Report No.: AIAA-1983-0037; 1983.
23. Chang KS, Choi CJ. Separated laminar natural convection above a horizontal isothermal square cylinder. *Int Commun Heat Mass Transfer* 1986;**13**(2):201–8.

Lu Haibo received his BE, MS and Ph.D. degrees from National University of Defense Technology in 2002, 2008 and 2012 respectively, and then became a teacher in Nanjing Artillery Academy. His current research interests are thermal analysis and control of high speed vehicles and its power system.

Liu Weiqiang is a professor at College of Aerospace and Material Engineering, National University of Defense Technology. He received the Ph.D. degree from the same university in 1999. His main research interests are thermal analysis and control of high speed vehicles and its power system.